COMPUTATIONAL STUDY OF FUEL SPRAY STRUCTURE

MOHD HILMI BIN MOHD ZIN

Thesis submitted in partial fulfillment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITY MALAYSIA PAHANG

JUNE 2013

ABSTRACT

This thesis deals with the study of fuel spray structure via computational (simulation) method. The main objective of this thesis to perform a computational study of pure gasoline fuel sprays structure development where it covers to parts; to determine the pure gasoline fuel spray angle and spray penetration depth characteristics using sing-hole port fuel injector (PFI) and to determine the impact of different injection pressure on the spray structure of pure gasoline fuel. The spray simulations are done completely by using Computational Fluid Dynamics (CFD) ANSYS CFX software with three nozzle tip diameter; 0.2mm, 0.3mm and 0.4mm. The Computational Aided Design (CAD) model for each nozzle was drawn using the SolidWorks software, the nozzle is attached with 110mm bore and 125mm stroke combustion chamber. In the ANSYS CFX software, the ready CAD model is imported into the design modeler and under goes meshing process with fine relevance center, 4×10^{-5}m min size, 4×10^{-3}m max face size and 8×10^{-3}m max size. There are three types of boundary conditions applied to the meshed geometry model, the first is inlet boundary condition with various injection pressure of 100bar, 150bar, 200bar and 250bar. Opening boundary condition is then place at the combustion chamber with atmospheric pressure value that is 101325Pa and the third boundary condition is wall. The iteration calculation is solved until the convergence approached to the desired residual value and the result is obtained and analyzed. The first comparison made is between penetration depth versus injection pressure and the other is between spray angle versus injection pressure, the results are then compared between nozzle diameter for each injection pressure. The results show that as the injection pressure increased, the penetration depth is also increased as well as the spray angle. The conclusion has shown that the nozzle tip diameter is also effecting the overall spray structure because wider nozzle tip diameter will released more fuel quantity compared to the smaller nozzle tip diameter.

ABSTRAK

Tesis ini adalah berkaitan dengan kajian struktur semburan bahan api melalui kaedah pengiraan (simulasi). Objektif utama projek ini adalah untuk melakukan kajian pengiraan pembangunan struktur semburan bahan api petrol tulen di mana ia meliputi bahagian-bahagian berikut; menentukan ciri-ciri sudut semburan dan kedalaman semburan bahan api petrol tulen dengan satu-lubang port penyuntik bahan api (PFI) dan menentukan kesan tekanan suntikan yang berbeza pada struktur semburan bahan api petrol tulen. Simulasi semburan dilakukan sepenuhnya dengan menggunakan Computational Fluid Dynamics (CFD) perisian ANSYS CFX dengan tiga diameter muncung yang berbeza; 0.2mm, 0.3mm dan 0.4mm. Model Computational Aided Design (CAD) untuk setiap muncung telah dilukis dengan menggunakan perisian SolidWorks, muncung telah dilukis bersama kebuk pembakaran berukuran 110mm diameter dan 125mm strok. Dalam perisian ANSYS CFX, model yang telah siap CAD diimport ke dalam reka bentuk pemodel dan melalui proses penjaringan dengan pilihan pusat relevan yang baik, 4×10^{-5}m saiz minimum, 4×10^{-3}m saiz muka makimum dan 8×10^{-3}m saiz maksimum. Terdapat tiga jenis keadaan sempadan yang digunakan terhadap model geometri, yang pertama adalah keadaan sempadan masuk dengan pelbagai tekanan suntikan seperti 100bar, 150bar, 200bar dan 250bar. Keadaan sempadan pembukaan kemudian meletakkan di kebuk pembakaran dengan nilai tekanan atmosfera, 101325Pa dan keadaan sempadan ketiga ialah dinding. Pengiraan lelaran diselesaikan sehingga penumpuan nilai lelaran mendekati nilai baki yang dikehendaki dan keputusan pengiraan diperoleh dan dianalisis. Perbandingan pertama yang dibuat adalah diantara kedalaman semburan berbanding tekanan suntikan dan perbandigan diantara sudut semburan berbanding tekanan suntikan, keputusan pengiraan jiga dibandingkan diantara diameter muncung dan setiap tekanan suntikan. Keputusan menunjukkan bahawa apabila tekanan suntikan meningkat, kedalaman semburan dan sudut semburan juga meningkat. Kesimpulannya menunjukkan bahawa diameter muncung juga memberi kesan terhadap keseluruhan struktur semburan kerana semburan muncung diameter yang lebih luas akan dikeluarkan kuantiti bahan api yang lebih banyak berbanding dengan muncung diameter yang lebih kecil.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

$\sum\dot{Q}$	Net rate of heat addition to the fluid
$\sum\dot{W}$	Net rate of work done by surface forces on the fluid
C_8H_{18}	Gasoline
C_c	Empirical constant
CO_2	Carbon Dioxide
d	Nozzle injector exit diameter
F_i	External body
H_2O	Water
L_c	Liquid core
N_2	Nitrogen
O_2	Oxygen
ρ	Fluid density
ρ_G	Gas densities
ρg_i	Gravitational body force
ρ_L	Liquid densities
t	Time interval
τ_{ij}	Stress tensor
V	Velocity at any point in the flow field
u	Velocity at x-axis direction
v	Velocity at y-axis direction
w	Velocity at z-axis direction

LIST OF ABBEREVIATIONS

CAD	Computational Aided Design
CFD	Computational Fluid Dynamics
PFI	Port Fuel Injector

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

The studies about fuel spray structure for both non-combustion and combustion process have been made multiple times, where most of the research focusing the dilute spray medium which is distant from the nozzle injector exit and the initial dispersed flow of the droplet breakup using common method such as observation, calculation, simulation and modeling are reasonably important due to tiny liquid volume fractions (Faeth, Hsiang and Wu, July 1995). Now a research is made to study the spray structure of a fuel focusing the whole medium, starts from the nozzle injector exit or known as dense spray medium until the dilute spray medium including the droplet features depend on various spray pressure using computational method.

General aspects that need to be include in this studies are divided into four categories, the first aspect is the spray structure in the dense spray medium where the spray liquid still not distributed right before and after the nozzle injector exit in order to investigate and define the initial spray properties such as liquid viscosity, pressure, temperature and volume (Faeth, Hsiang and Wu, July 1995). The second aspect is the properties of primary breakup, it is the initial conditions for the dense sprays medium including both spray structure properties and the hardware properties such as the nozzle injector exit. Every single thing such as pressure, viscosity and more are important and has the potential to influence the structural characteristics of the spray (Faeth, Hsiang and Wu, July 1995).

The third aspect is the properties of secondary breakup which will closely related with the rate controlling process of dense spray medium and structural characteristics of the droplet which also related with the rate controlling process of dilute spray medium. Each characteristics of spray structure like spray distributions, position of spray structure and spray tip penetration are essential to observed in order to gain the perfect outcome (Faeth, Hsiang and Wu, July 1995). The last aspect is the properties of the droplet characteristics at the end of spray distribution such as droplet sizing, where the spray already passed through the nozzle injector exit, dense spray medium and dilute spray medium (Faeth, Hsiang and Wu, July 1995).

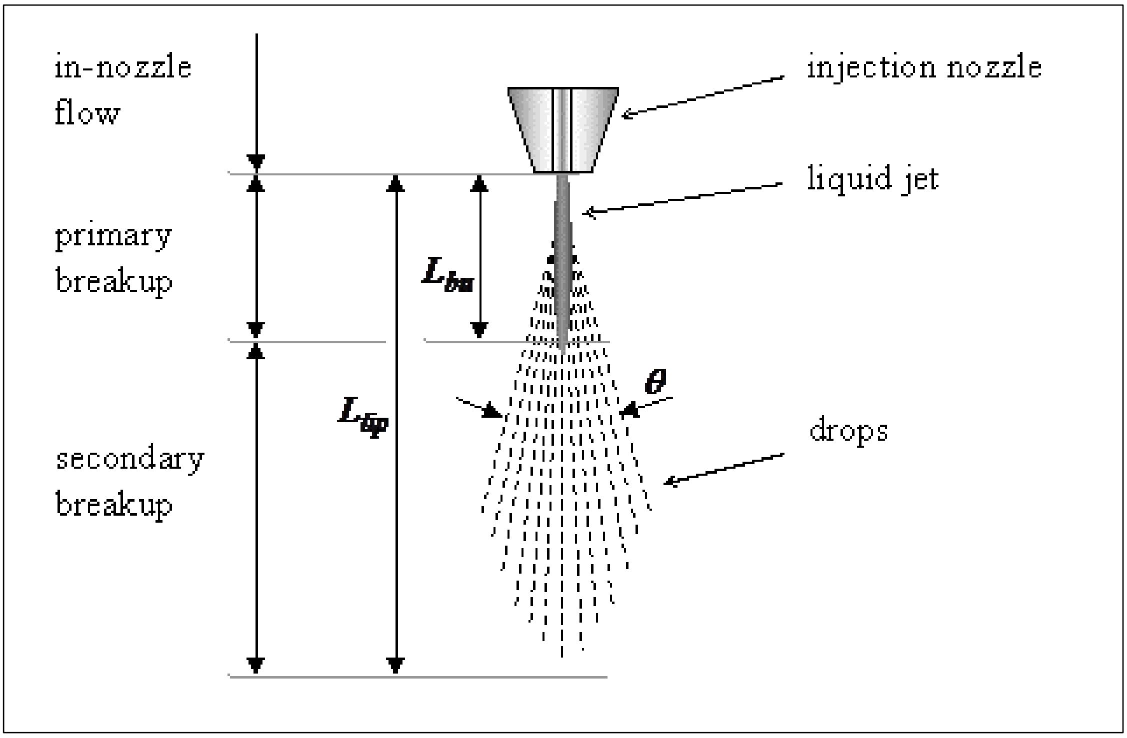

Figure 1.1: Main physical parameters on spray structure

Figure 1.1 shows the main physical parameters on spray structure with the primary breakup and the secondary breakup, it illustrate a measured spray characteristics that are basically been classified into two categories. The first one is called as "Macroscopic Characteristics" that focus on both primary breakup and secondary breakup that containing the spray angle and spray tip penetration. The second one is called as "Microscopic Characteristics" which focus only on the secondary breakup containing droplet distribution (diameter), droplet velocity, air-fuel ratio and so forth. A real experiment would be more reliable because it is exposed to the real world conditions such as pressure, temperature, humidity and any other properties that might be affecting the spray structure development.

1.2 PROBLEM STATEMENT

Nowadays the development in automobile engine is sharply improved with the emergence of bio-fuel engine system, hybrid car system and many more (Anand, Madan Mohan & Ravikrishna, 2012). In order to know how pure gasoline can produce the maximum power output or result that can be matched with the advanced system, a computational study of fuel spray structure is essential. Spray fuel structure of pure gasoline is an important factor to be study and investigate due to differences in characteristics such as injection pressure, droplets size distribution, spray progression, position of spray structure and spray tip penetration may result in different power output of an internal combustion engine (Schmehl, Maier & Wittig, 2000). The competition between advanced fuel and pure gasoline in power output value is vastly intense due to differences chemical substances in each burning fuel.

Different fuel will provide different power output result, hence the computational study of fuel spray structure of pure gasoline is important to identify the characteristics of spray development, droplet size distribution and many more in order to increase the power output of an engine. Any factors that might be affecting the spray structure such as spray angle, depth, type of nozzle and injection pressure will be included in this study. The spray itself must obey air-fuel ratio to generate maximum heat energy that can be transform to mechanical work. Another problem why computational study on spray structure is essential because of incomplete burning of fuel will result in less energy for the mechanical work and at the same time will affect the condition inside the engine (Rossella Rotondi & Gino Bella, 2005).

1.3 OBJECTIVES

The main objective of this research is to perform a computational study of pure gasoline spray structure development where it will cover two parts:

i. To determine the pure gasoline fuel spray angle and depth characteristics using single-hole port fuel injector (PFI).

ii. To determine the impact of different injection pressure on the spray structure of pure gasoline fuel.

1.4 SCOPE OF STUDY

This project is focus on computational study of pure gasoline fuel spray structure development using suitable software that is Computational Fluid Dynamics (CFD). The entire computational study will be performing using several different amount of injection pressure and using a single-hole PFI. To complete this project, the actions are required:

i. Study of spray angle and depth of gasoline fuel by using single-hole PFI.
ii. Study the effect of different injection pressure on spray structure which is in the range of 100bar to 250bar.
iii. Study the result of analysis from the simulation done.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Improving modern internal combustion engine efficiencies by increasing or decreasing the pressure levels of the combustion processes require sophisticated combustion concept and analysis method. The principles of modern internal combustion engine are the strategic characteristics to inject the liquid fuel and to mix it with the flow of compressed air. In order to study and understand the fuel spray structure, computational study is required than depending on previous experiments and researches (Schmehl, Maier & Wittig, 2000).

2.2 DENSE AND DILUTE SPRAY STRUCTURE

Dense spray structure is a part of spray structure where it covers the medium between the nozzle injector exit and the dilute spray medium, a sketch of spray structure near the nozzle injector exit is illustrated in Figure 2.1. It is also a medium where the spray structure will come together and mix with the gas phase inside the combustion chamber. During practical combustion processes, the atomization of the spray breakup is the most important for the rapid mixing of the fuel and the oxygen where both of them existed in liquid and gas phase (Reitz & Bracco, 1982). Dense spray medium consist of two main multiphase flows, the first multiphase is the liquid core where the liquid are mostly does not mix with the oxygen. The second multiphase is the dispersed flow region where the liquid is already mixed with the oxygen and atomization or spray droplet had been developed.

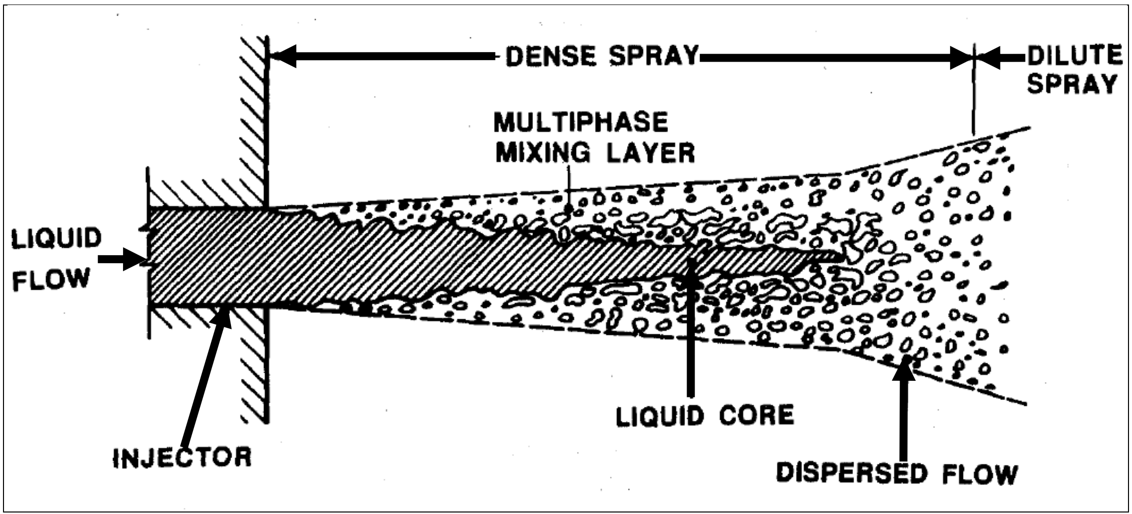

Figure 2.1: Sketch of spray structure near the nozzle injector exit

Source: Faeth, Hsiang and Wu, July 1995

The liquid core is similar to the potential core of a single phase nozzle injector exit although it is generally much longer. The Eq. (2.1) below represents the length of the liquid core, L_c, where d is the nozzle injector exit diameter, ρ_L and ρ_G are liquid and gas densities and C_c is an empirical constant in the range of 7-16. This visualizes that L_c/d in the range of 200-500 for a typical spray at atmospheric pressure, with this ratio generally being inversely proportional to the square root of pressure. Hence, liquid core is most visible feature of round pressure-atomized spray (Chehroudi, 1985).

$$L_c/d = C_c(\rho_L/\rho_G)^{1/2} \tag{2.1}$$

Dispersed flow region take place at the end of the liquid core, it involves a developing multiphase mixing layers between the liquid and the gas, followed by a multiphase layers that evolves into spray droplets in dilute spray flow. It is a region which connected the dense spray medium and dilute spray medium, that shows the mixing of both liquid and gas had been happen. The dense spray medium generally related with the liquid core even though it is not totally accurate because at the end of dense spray medium is a dilute spray medium while the initial liquid core’s flow has a large liquid volume fractions. The properties and existence of the dense spray medium are very dependent toward liquid flow properties such as disturbance levels and turbulence levels at the nozzle injector exit.

2.3 SPRAY ANGLE AND SPRAY TIP PENETRATION

In definition, spray angle is the angle of opening of dispersed fluid flow that experienced transformation from laminar flow to turbulent flow under certain condition. Spray angle is determined by many factors such as opening dimension, pressure, viscosity and so forth, it is known that the spray angle does not holds for the entire spray propagation. It tends to collapse or diverge as it moves away from the nozzle tip. An assumption has been made where the spray angle will remain constant throughout the spray distance travelled, but in actual situation the angle will not remain constant throughout the spray distance travelled. Figure 2.2 illustrate the difference between actual and theoretical spray angle. Spray tip penetration or spray depth is the total length of fluid spray structure between the nozzle tip and the end of spray propagation, the distance is determined by the injection pressure and also influenced by the opening dimension.

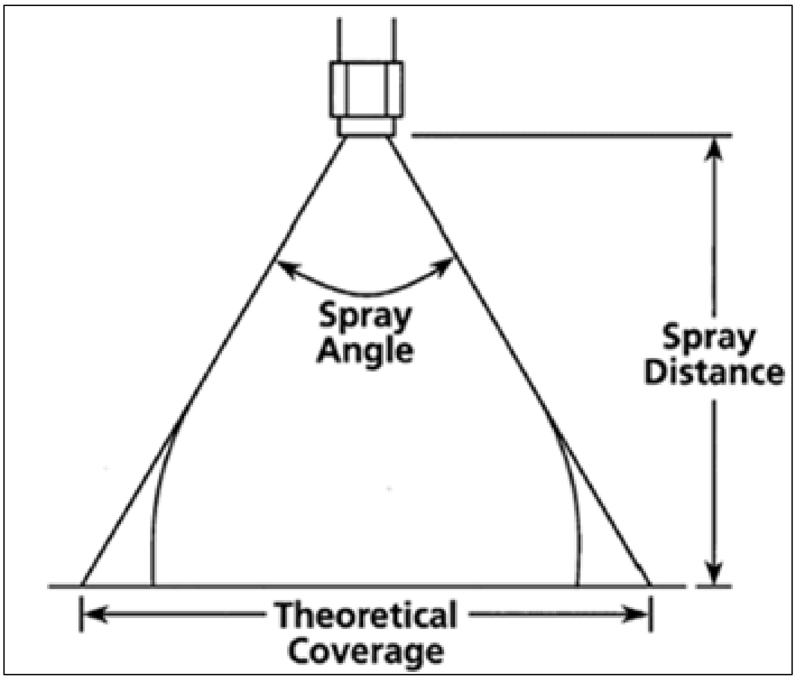

Figure 2.2: Illustration of spray angle

2.4 FLOW STRUCTURE OF DENSE SPRAY MEDIUM

In order to evade the significant effect of the degree of turbulence development at the nozzle injector exit, further information about dense spray properties will be limited to condition where there is fully-developed turbulence flow at the nozzle injector exit (Tseng, 1992). Time-average liquid volume fractions were predicted at various pressure levels, this prediction are based on the turbulence model under the locally-homogenous flow (LHF) approximation. For a complete explanation of this model prediction, the relative velocities between the phases are assumed to be very small in comparison to the mean flow velocities (Ruff, 1989).

Estimated liquid volume fractions near the nozzle injector exit are unity followed by rapid reduction of the liquid volume fractions due to spray breakup development. As the pressure increases, the initial reduction of liquid volume fractions becomes smaller and indicating faster mixing rates at higher ambient gas densities (Ricou & Spalding, 1961). With this condition, LHF predictions generally are good because separated flow effect due to relative velocity differences between liquid and gas are not very prominent when the flow is mostly in liquid phase. Although the variation on liquid volume fractions suggests a relatively short liquid core, this is not completely related in terms of mixture fractions.

2.5 PRIMARY BREAKUP

In primary breakup, the most important process spray structure development is the droplets formation near the liquid surfaces because it initiates the atomization process, controlling the liquid core length and provides the initial condition of the dispersed flow region. Due to problems of observing the primary breakup in the dense spray medium and effects of secondary breakup, the current information and understanding of primary breakup is limited. Other than that, the effects of flow development and liquid disturbances such as turbulence at the nozzle injector exit provide an unusual large impact on the primary breakup properties. With the present of pulsed holography technique, it have provided a chance to observe the properties of dense spray medium for making the progress move forward and at the same time gaining a better understanding of primary breakup process (Wu & Faeth, 1993).

2.5.1 Onset of Breakup

All spray properties that have been established by the past studies including criteria for the onset of breakup are strongly affected by the degree of flow development and the disturbance such as turbulence at the nozzle injector exit. Based on early studies of pressure atomization shows that both mixing rates and atomization quality are not the same for laminar and turbulence flow at the nozzle injector exit (Lee & Spencer, 1933). Further studies conclude that turbulence generated in the flow has a small effect on the spray droplet properties. After that, another studies shows that spray breakup could be suppressed entirely for super-cavitations flows where liquid jet neither separates from the injector route wall near the end of the contraction section and does nor reattach (Karasawa, 1992).

Other studies also have discover that liquid phase flow properties have dominated observations of primary breakup in pressure-atomized spray and the aerodynamics effect does not very crucial at the liquid or gas density ratios at normal pressure and temperature. The breakup of the liquid jet in an air at atmospheric pressure was related with the presence of turbulent boundary layers along the injector route walls near the nozzle injector exit (Hoyt & Taylor, 1977). Other than that, large changes in the aerodynamics environment including both counter-flowing and co-flowing air result a small effect on the breakup properties. In a reality, the actual properties of the turbulent boundary layers along the injector route walls will be ignore in any experimental condition (Hoyt & Taylor, 1977).

2.5.2 Breakup Outcomes

Once the condition for the onset of turbulent primary breakup is determined, the breakup outcomes that cover the variation of spray droplet velocity and size distribution with increasing length from the nozzle injector exit will be review. After the turbulent primary breakup, the spray droplet sizes satisfy the universal root normal distribution and the spray droplet distribution is uniform. According to the previous turbulent primary breakup experiment, there are three types of turbulent primary breakup (Wu & Faeth, 1993):

1. Non-aerodynamics turbulent primary breakup.

2. Aerodynamically-enhanced turbulent primary breakup, observed at the onset conditions.
3. Aerodynamic turbulent primary breakup, which involves unification of turbulent primary and secondary breakup.

The result from the previous experiment shows that the boundaries of these turbulent breakups are fixed by the liquid or gas density ratio, the breakup times used to determine types of turbulent primary breakup were based on the mean diameter of the spray after the primary breakup or after the primary breakup stage of combined primary and secondary breakup for condition outside the onset of breakup.

A major issue still uncover involves primary breakup of non-turbulent liquids and the relevance of the classical primary breakup theories (Taylor, 1963 and Levich, 1962). Results show that it is hard to observe the non-turbulent primary structure. The main obstacles are effects of liquid disturbances such as turbulent, the invasion of secondary breakup and weak aerodynamics effects for most liquid at atmospherics temperature and pressure.

2.6 SECONDARY BREAKUP

Based on the previous considerations of the spray structure of dense spray medium, the secondary breakup is essential with its effect on the spray droplet size distribution as the flow movement approaching the dilute spray medium. As reviewed before, primary breakup at the surface of the liquid core developed spray droplets that ate unstable to the formation and development of the secondary breakup. Other than that, both typical power and propulsion systems of high-pressure combustion involves situation where the surface tension of spray droplets becomes small due to the liquid surface move towards the thermodynamics critical point.

In previous findings, there have two limitations of define disturbances that cause the deformation and spray breakup droplets. The first limitations is the shock wave disturbances that provide changes in the ambient environment of a spray droplets at the end of the primary breakup, while the other limitations is the steady disturbances of freely-falling spray droplets in spray drying processes or in rainstorm. The shock wave disturbances effects have become the major attention and approximate the secondary breakup environment in the dense spray medium.

2.6.1 Deformation and Breakup Movements

Many studies and researches have made an assumption on the conditions and definitions for the onset of various deformation and breakup movements of spray droplets subjected to shock wave disturbances. When the liquid viscosity effects are relatively small, the observed breakup movement at the onset of breakup has been termed as ‘bag breakup’ illustrated in Figure 2.3. This ‘bag breakup’ is the deflection of the spray droplets into a thin disk normal to the flow path and the deformation of the middle of the disk (Wierzba & Takayama, 1988).

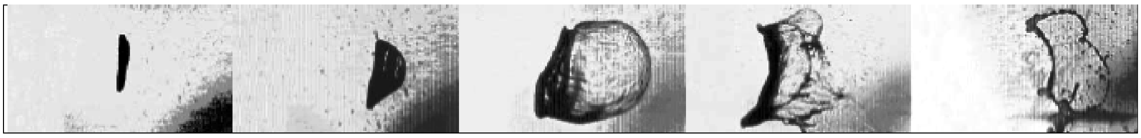

Figure 2.3: Spray droplets deformation and ‘bag breakup’

Source: Schmehl, Maier & Wittig, 2000

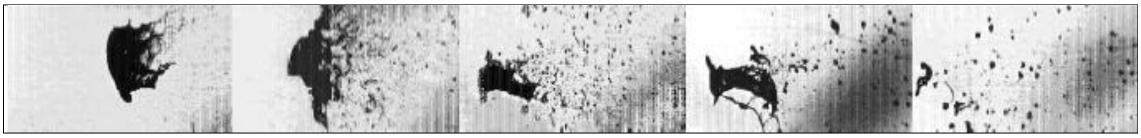

Figure 2.4: Shear breakup

Source: Schmehl, Maier & Wittig, 2000

The observations of shear spray breakup as illustrated in Figure 2.4 have been made at high relative velocities where the shear spray breakup experienced a deflection of the edge of the disk in the downward path, deflection of the middle of the disk and the stripping of spray droplets from the edge of the disk. The conversion between the ‘bag breakup’ and the shear breakup movement is a complex mixture where this complex breakup mechanism can only be observed at high relative velocities and known as ‘catastrophic breakup’ (Reinecke & Waldman, 1970).

2.6.2 Breakup Dynamics

The discussion about deformations and spray breakup movement transitions prioritize the importance of breakup times and identify its characteristics when the liquid viscosity forces are in huge comparison to the force of surface tension. The period before the onset of the spray breakup is the period where the spray droplets experienced significant deformation, the drops are initially drawn into a flat shape because of the presence of the relative motion of the gas phase. Certain researches have summarized a relatively large data base of maximum spray breakup droplets deformations for steady disturbances (Hsiang & Faeth, 1992).

2.6.3 Breakup Outcomes

Secondary breakup can be treated using jump condition with the assumption of spray breakup times and distances are relatively small compared to the characteristics of dense spray medium. To fulfill this approach, information about spray breakup droplets size and velocity distributions right after the secondary breakup take place are essential. Measurement information for the ‘bag breakup’ movement is limited to provide enough guidance about the spray breakup droplets sizes as the result obtained from the secondary breakup (Gel’fand, 1963). Further research used the pulsed holography technique and obtained a complete description of the secondary breakup outcomes for shock wave disturbances conditions (Hsiang & Faeth, 1993).

The secondary breakup in the dense spray medium is not properly represented by jump conditions at the high pressure surrounding of multiple practical spray combustion processes. Under such obstacles, the secondary spray breakup should be assumed as a rate process. Other than the deformations and spray breakup movements, existing information about the secondary spray breakup is still limited and it show clearly that additional study or research is essential in order to gain better understanding of secondary spray breakup properties for practical combustion processes.

2.7 COMPUTATIONAL FLUID DYNAMICS (CFD) SOFTWARE

Computational fluid dynamics (CFD) is a modern analysis process that used numerical and algorithm method which allows a computational model representing the physical system to be built or studied and uses computers to simulate fluid flow dynamics. CFD itself raise the head as a useful tool to reduce cost and time waste by computational method compared with costly and more time consumed experiments to produce a much better result and design. Experimental data is also required for input in CFD simulations for example the flow type and boundary conditions properties. When fluid flow model is applied to this virtual prototype, the CFD software application is capable to predict the outputs of the fluid dynamics.

Other than that CFD also predict the transfer of heat, mass, phase change, chemical reaction, mechanical movement, stress or deformation of related fluid structures and associated phenomena such as chemical reactions by means on computer based simulation (Baris Guler & Rizwan Ali, October 2004). Both compressible and incompressible fluid flows can be combined with specific properties and parameters, all the simulations can be done using 2D and 3D flows (www.cosmol.com). Advantages and benefits of CFD simulation are listed below:

- 3D surface and solid modeling
- Simulation, visualization and analysis of the fluid flow
- Full analysis report including integrated quantities
- Capable to shows result animations and pictures of fluid flow field
- Alterations done in the 3D model are associative with mesh
- Quick recalculation
- Many operating conditions can be calculated with same analysis model

In order to determine the spray structure characteristics such as spray angle and depth of gasoline fuel, CFD CFX software will be used for simulation and get the result. The parameter that will be examined by using this CFD software is different injection pressure range from 100bar to 250bar. This parameter will be the manipulated variable and the final simulation results are capable to determine the spray angle and depth. Another objective that needs to accomplish is to determine the impact of different injection pressure on the spray structure development. From the final simulation result